

Numerical modeling of airflow over column of vehicles using Ansys® package

Władysław Hamiga^{1,*} and Wojciech Ciesielka¹

¹AGH University of Science and Technology, Mickiewicza Av. 30 30-059 Cracow, Poland

Abstract. Growing needs in transportation determinate systems development which improve efficiency of travel and reduce harmful influence to environment. Computational Fluid Dynamic (CFD) uses numerical analysis to find optimal solutions in terms of chosen objective function. Time save and reduce costs for experiments on prototypes are one of the advantages of this method.

The aim of this research is to analyze airflow around different motor vehicles which are moving together in the same direction. To reduce fuel consumption and, at the same time, decrease negative influence to environment, the primary target was reducing total drag force during a ride. The vehicles were set in a column - one after another. In this work considered three types of vehicles: Car, Van and a Truck. Presented vehicles were organized into appropriate groups, creating different configuration. Additional parameter in simulation was distance between vehicles. Simulations of singular vehicles were also done. It allows to evaluate influence of moving vehicles in a column for generated drag force. Described traffic situation were modeled and numerically calculated using ANSYS® package. The purpose of this work was to assess the impact of the distance between vehicles, in a given configuration, for generated drag force.

1 Introduction

More and more studies concentrate on realistic situation on road, where information about aerodynamic characteristic such as drag and lift of singular vehicle is not enough. Researchers investigation going around issue of motion of group of vehicles traveling together (platoons). The advantages of forming vehicles in convoys is both. Increasing of fuel efficiency and improving driver comfort and safeness. Fuel efficiency increase because of reducing pressure and friction drag force, which is result of moving in environment of surrounding fluid (air). Reduce of air drag force is significant aspect in saving energy of moving vehicles. Estimating, the air drag is generating about 52% of total resistance in motion at 100km/h for large, heavy tracks [8]. For this reason aerodynamic characteristics of vehicles are very important.

Study in full-scale models are performed on real vehicles under normal operating conditions. However, road tests are sensitive to ambient conditions such as weather, or they

* Corresponding author: hamiga@agh.edu.pl

are disturbed by vehicles which are not participating in the study. Another way are tests realized in wind tunnel with controllable and known boundary conditions. Experimental study in wind tunnel giving good results but it is very expensive, if good representation of reality is necessary (moving track, rotating wheels, full-scale geometric model, shape's details). Therefore this research method is commonly use in validation process, where geometry and boundary condition are simplify and model can be made in scale. Data from experiment is useful to create numerical model, which is used in Computational Fluid Dynamic (CFD) analysis. Numerical calculations are beneficial for saving time and reducing cost of study.

The study of aerodynamic interaction between vehicles has been investigated for years. Experiments are different between each other in terms of the applied research method, vehicle geometry, and their configuration. Interaction of two cars moving on highway was analyzed by Watkins et al. [10]. Special consideration is emphasis on spacing between vehicles. The experimental studies have been conducted on two Ahmed bodies with backlight angle 30 degree. Experiment was executed in close-jet and fixed-ground type of wind tunnel. Authors prove drag force reduction for close spacing and significant changes in lift. Experimentally study of vehicle platooning in wind tunnel is hard because of huge size of model. To compensate model scaling and increase measurement accuracy, the wake generators was used by Hesham et al. [5]. Further CFD analysis was conducted using STAR-CCM+ package. Behavior of flow pattern around vehicle, during acceleration and overtaking maneuver, was described by Manimaran [6]. In simulations, geometric model of two cars is Ahmed body with backlight angle 25 degree. Author using k- ϵ model of turbulence in OpenFOAM software. Schito et al. [7] conducted his research in two stages. At first, CFD simulation was performed on Ahmed bodies with backlight angle of 0 and 30 degree. In second part the Ahmed bodies have been replaced by simplified vehicles shapes of compact car, sedan, van and a truck. In both cases, three different spacing was considered. Calculation of singular vehicle has been performed too.

The aim of this work is to reduce drag force of moving column of vehicles. Three type of vehicles was considered: Car, Van and a Truck. Decrease of acting drag force is effect of appropriate configuration and spacing. Appropriate drive in column will reduce fuel or electrical energy consumption, and will have beneficial effect on environment.

2 Methodology

In order to study aerodynamic parameters of moving column of vehicles, a series of CFD simulation has been conducted. Six configuration of cars was investigated. In each of configuration it was considered three type of vehicles. Spacing between vehicles is from 0.3 to 100 meters. The condition of simulation is set to steady-state. Therefore, fluid properties like velocity, pressure or mass flow rate do not change over time. The volume meshes were built using hexahedral cells. Elements around vehicle's geometric has been applied using inflation layer. Inflation layer is 20 cell height with corresponding y^+ value near 35 [9]. As turbulence model, in simulation, was used k- ω Shear Stress Transport (SST). Output parameter of CFD simulation is drag force, acting on each of the vehicles individually. Forces were recalculate to dimensionless drag coefficients according to equation:

$$C_d = \frac{2F_d}{\rho A v^2} \quad (1)$$

where :

- F_d – acting drag force [N],
- ρ – air density [kg m^{-3}],
- A – frontal area of the vehicle [m^2],
- v – relative fluid velocity [m s^{-1}].

Grid validation was realized comparing CFD simulation of airflow over Ahmed body, with experimental data from the literature [1], [4]. Ahmed body with backlight angle 25° , without stilts was considered. Based on the conducted simulation tests, which were compared with the data in the above-mentioned literature, it was shown, that the calculated drag coefficient error did not exceed 5%.

All of CFD analysis were conducted in ANSYS FLUENT, while mesh was prepared in ICEM software (Fig.1)



Fig. 1. Hexahedral mesh. Vehicle configuration: Truck-Car-Van and spacing 0.3m

2.1 Object of research

Geometry models of vehicles used in this study, are simplified to the shape of rectangular cuboid. Size of the vehicle was considered as main criterion. Therefore, differences between three types of vehicles are only in height, width and length. Individual dimensions are given in the Table 1.

Table 1. Vehicles dimensions

Vehilce	Height H [m]	Width W [m]	Length L [m]
Car	1.49	1.77	4.55
Van	2.5	2.05	6
Truck	2.8	2.48	14

Clearance between vehicles and the road is set to 0.15m.

2.2 Boundary conditions

Domain of CFD analysis is rectangular cuboid with dimension 20m x 20m x 125 – 325m (last dimension depends of spacing between vehicles). Geometric model of domain is shown at Fig.2. To reduce number of cells, symmetry condition is set up on one of the plane, so only half of model is considered. Velocity of fluid is define as 25m/s at the inlet. Road is set up as "moving wall" with absolute velocity 25m/s. Elements such as the surfaces closing the area of simulation (top and side), as well as the surfaces of the vehicles

(vehicle 1, vehicle 2, vehicle 3) has been set as "walls" with zero normal velocity. The boundary condition at the outlet is define as the ambient pressure.

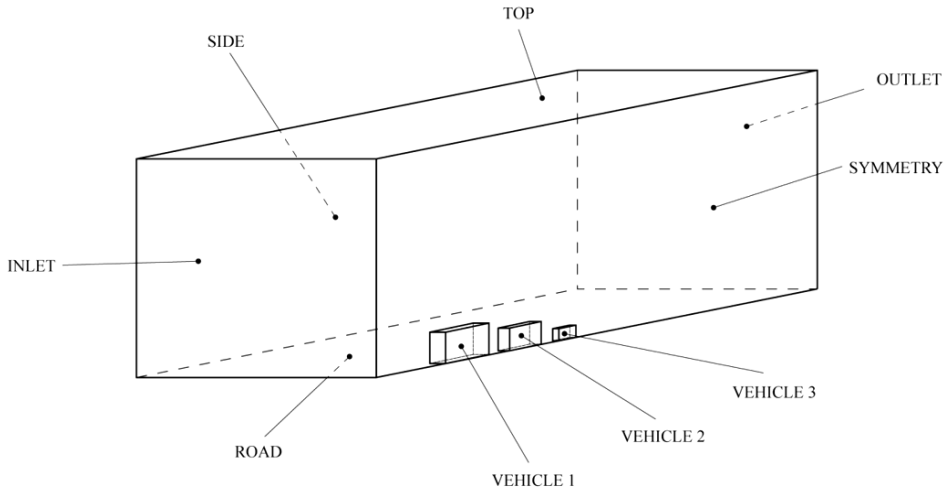


Fig. 2. Model description

3 Results and discussion

The study of aerodynamic interaction between vehicles has been investigated for six different configuration. The column of vehicles consisted of Car, Van and a Truck. Relationship between spacing and drag coefficient for all configurations is presented on charts (Fig.4 - Fig.9). Drag coefficient is calculated for each of the vehicle individually.

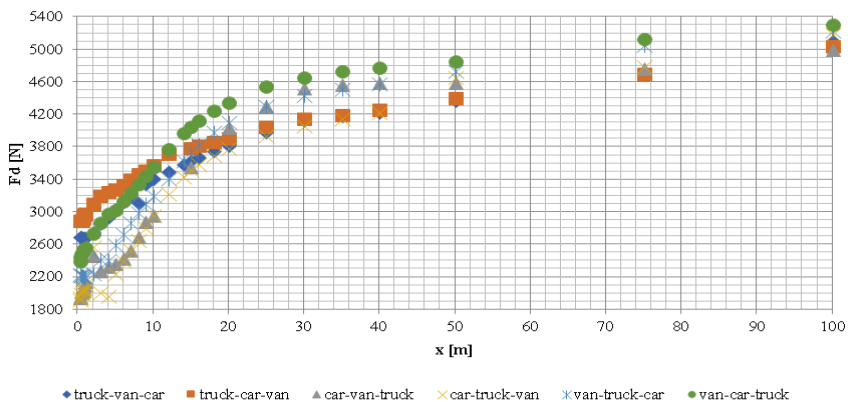


Fig. 3. Total drag force as a function of spacing for all configurations

Drag force presented on Fig.3 is the sum of the forces generated on individual vehicles (drag force of column). Drag force reduction is observable in all configurations. However, significant changes are only on second and third vehicle (Fig.4 - Fig.9). Reduction, in this case, in drag force reaching up to 66% for Car-Truck-Van configuration. It should be noted that distance of 0.3 meters is dangerously small, therefore study on higher vehicle spacing was conducted. For spacing 1 to 18 meters, the most beneficial configuration is with Car on lead. Above 18 meter, in configuration Car-Van-Truck, drag coefficient of Van and Truck is rapidly increasing (Fig.6). Decrease of drag coefficient is also visible for leading

vehicles. However, it is only for spacing below 20 meter for Car (Fig.6, Fig.7) and below 12 meters for Truck (Fig.4, Fig.5). Configurations with Vans on lead do not show any benefits for them in drag coefficient reduction (Fig.8, Fig.9).

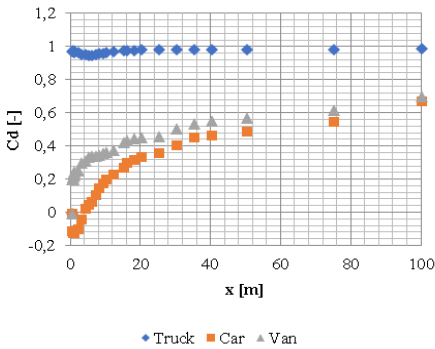


Fig. 4. Drag coefficient as a function of spacing. Configuration: Truck-Car-Van

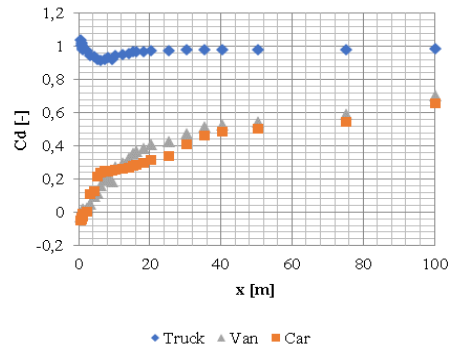


Fig.5. Drag coefficient as a function of spacing. Configuration: Truck-Van-Car

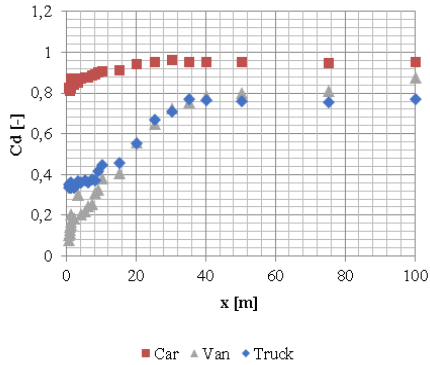


Fig. 6. Drag coefficient as a function of spacing. Configuration: Car-Van-Truck

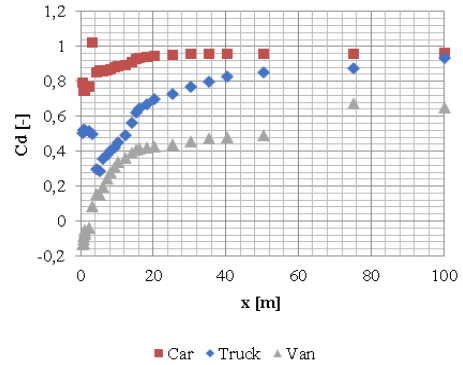


Fig. 7. Drag coefficient as a function of spacing. Configuration: Car-Truck-Van

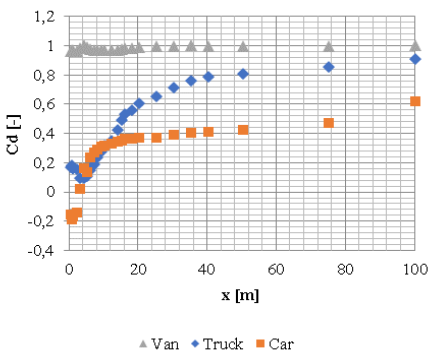


Fig. 8. Drag coefficient as a function of spacing. Configuration: Van-Truck-Car

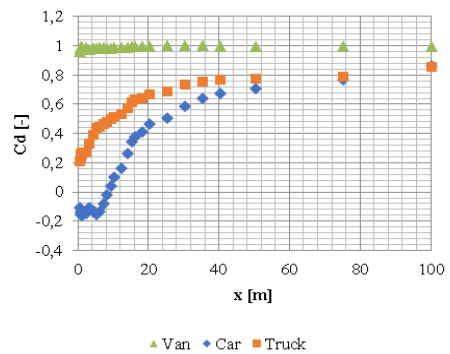


Fig. 9. Drag coefficient as a function of spacing. Configuration: Van-Car-Truck

The CFD simulations of singular vehicles were also conducted. For increasing distance between vehicles, drag force tends to the value of singular vehicle. The value of drag coefficient going to coincide (Fig.4 - Fig.9).

With objective function to reduce total drag force of whole column, the most beneficial configuration is Car-Truck-Van (Fig.3). The shape of column, in this configuration, approximately, is similar to water drop.

Velocity distribution on symmetry plane is presented for distance between vehicles equals 6 meters (Fig.10 - Fig. 15). Comparison of velocity is shown for all configurations. Velocity distribution legend for Fig.10 - Fig. 15 is presented in Fig. 16. Wake behind column of vehicles is observable in all configurations and it is spread for whole domain.

4 Conclusion

In the paper was presented CFD study of airflow over the column of vehicles using ANSYS package. Drag coefficient, as a function of spacing, has been investigated for six different configuration. The validation process was executed on Ahmed body with backlight angle 25 degree and no stilts. Difference between calculated drag coefficient from simulation, and drag coefficient from literature, did not exceed 5%. Dependence of drag force and drag coefficient as a function of spacing was demonstrated on charts. Drag coefficient for small spacing (up to 1 meter) was significant reduced in all configurations, but only on second and third vehicle in the row.

Further study on aerodynamic parameters of convoy could be realize with more vehicles. Geometry of vehicles should be improve to achieve more accurate value of drag forces and drag coefficients. To reduce time of simulation, geometry in this work, was simplified. Presented numerical model will be use for investigation of acoustic emission, in aspect of its influence to the environment. The results presented in this work have a universal character and may be used to build intelligent acoustic environment management systems for boroughs, districts, cities and urban conurbations [2], [3].

References

1. Bello-Millan F. J., Makela T., Parras L., Pino C., Ferrera C.: Experimental study on Ahmed's body drag coefficient for different yaw angles, *Journal of Wind Engineering and Industrial Aerodynamics*, **157**, pp. 140-144, (2016)
2. Ciesielka W.: Management of environmental noise – Cracow example, *Polish Journal of Environmental Studies*, **16**, 3B, pp. 53–58, (2007)
3. Ciesielka W.: Management of environmental noise – the Cracow example, *Archives of Acoustics*, **32**, 4, pp. 983–994, (2007)
4. Guilmineau E.: Computational study of flow around a simplified car body, *Journal of Wind Engineering and Industrial Aerodynamics*, **96**, pp. 1207-1217, (2008)
5. Hesham M. Ebrahim, Robert G. Dominy, Pak S. Leung: Evaluation of Vehicle Platooning Aerodynamics using Bluff Body Wake Generators and CFD, *Students on Applied Engineering (ICSAE)*, pp. 218-223, (2017)
6. Manimaran R.: Aerodynamic Study of the Ahmed Body in Road-Situations using Computational Fluid Dynamics, *Fluids, Heat and Mass Transfer, Mechanical and Civil Engineering*, pp. 140-147, (2015)
7. Schito P., Braghin F.: Numerical and Experimental Investigation on Vehicles in Platoon, *SAE International Journal of Commercial Vehicles*, **5**, 1, pp. 63-71, (2012)
8. Schoon, R. E.: On-road Evaluation of Devices to Reduce Heavy Truck Aerodynamic Drag, SAE Publication, SAE Paper No. 2007-01-4294, (2007)

9. Simmonds N., Pitman J., Tsoutsanis P., Jenkins K., Gaylard A., Jansen W.: Complete body aerodynamic study of three vehicles, *Proceedings of WCX17: SAE World Congress Experience, 4 - 6 April 2017, Detroit, MI*, Paper number 2017-01-1529, (2017)
10. Watkins S., Vio G.: The effect of vehicle spacing on aerodynamics of representative car shape, *Journal of Wind Engineering and Industrial Aerodynamics*, **96**, pp. 1232-1239, (2008)